TITLE:

# Transition to turbulence in the Wake of a Bluff Body (Coherent Vortical Structures : Their Roles in Turbulence Dynamics)

AUTHOR(S):

Kuwahara, Kunio

CITATION:

Kuwahara, Kunio. Transition to turbulence in the Wake of a Bluff Body (Coherent Vortical Structures : Their Roles in Turbulence Dynamics). 数理解析研究所講究録 2000, 1121: 115-119

ISSUE DATE:

2000-01

URL:

http://hdl.handle.net/2433/63521

RIGHT:

京都大学学術情報リポジトリ KURENAI 紅
Kyoto University Research Information Repository

数理解析研究所講究録
1121 巻 2000 年 115-119



# Transition to turbulence in the Wake of a Bluff Body

ISAS Kunio Kuwahara (桑原 邦郎)

Impulsively-started flow past a bluff body is simulated by solving the unsteady incompressible Navier-Stokes equations at high Reynolds numbers. The separation and the transition to turbulence are investigated in the wake. The multi-directional finite-difference method with third-order upwinding is employed without using any turbulence model. Two examples are studied; the one is a sphere and the other is a complicated body called protoceratopus.

## I. Introduction

Flow around a bluff body is characterized by a strong vortex separation. This makes the flow turbulent in the wake. However, the mechanism of the separation leading to turbulence is not well understood because experimentally it is very difficult to see the very short period of transition. In this paper, the impulsively started flow around a bluff body is studied computationally at high Reynolds number and is visualized extensively to see the transitional flow phenomena to turbulence. The incompressible Navier-Stokes equations are solved by multi-directional finite difference method. Cartesian grid system with uniform spacing is used. Typical number of grid points is six million. The Reynolds number is 200,000.

Many of the high-Reynolds-number, turbulence simulations have been based on Reynolds-averaged Navier-Stokes equations using a turbulence model. Some use a large-eddy simulation based on a Smagorinsky-type model. However, a usual turbulence model or a large-eddy simulation is not suitable for high-Reynolds-number-flow computation especially transitional phenomena. There are some real direct numerical simulation in which most of the small-scale structure are resolved, but the computations can be done only at relatively small Reynolds numbers. We can not use enough grid points for high-Reynolds-number flows of practical interest. We have rather to use a very coarse grid system.

In many applications, large structures are most important and we usually are not interested much about in small structures. What we want to do is to capture the large-scale structure using a coarse grid system.

On the other hand, quit a few simulations (see Kuwahara, 1992), show that large structures of high-Reynolds-number, turbulent flow can be captured using relatively coarse grid, if the numerical instability, usually unavoidable for high-Reynolds-number-flow simulation, is suppressed. Most successful simulations in these approaches are based on the third-order upwind formulation (Kawamura and Kuwahara, 1984). An approach similar in philosophy but different in method is adopted by Boris et. al. (1992).

In the present paper, we simulated the transition to turbulence, which is impossible to compute by Reynolds averaged Navier-Stokes equations and very difficult by large eddy simulation.

## II. Computational method

The governing equations are the unsteady incompressible Navier-Stokes equations and the equation of continuity. For high-Reynolds-number flows, time-dependent computations are required owing to the strong unsteadiness.

These equations are solved by a finite-difference method. The numerical procedure is based on the projection method. Then, the pressure field is obtained by solving the Poisson equation.

All the spatial derivative terms are represented by the central difference approximation except for the convection terms. For the convection terms, the third-order upwind difference is used. This is the most important point for high-Reynolds-number computations.

For all the spatial derivatives, the multi-directional finite-difference method is used. For the temporal integration of the Navier-Stokes equations, the Crank-Nicolson implicit scheme is utilized. This scheme has second-order accuracy in time. These equations and the Poisson equation are iteratively solved at each time step by the successive overrelaxation (SOR) method with multigrid technique.

## III. Computational results

It is impossible to compute transitional flow by using turbulence model. Even large-eddy simulation can not handle this type of problem, because it assumes the flow is turbulent from the beginning. However, transition phenomena is very important at high-Reynolds number flow. In the present approach, this is not a special thing, we can compute any transitional flow with no special consideration. In Fig.1, development of turbulence behind a sphere is shown; the sphere has impulsively started from rest. Fig.2-4 show a flow around a ski jumper. This is a more streamlined body than the sphere. The turbulent area is much less. Fig.5 shows a flow around a protoceratopus. This is as bluff as a sphere, then the wake turbulence is very similar to the sphere. The visualization was done by showing the surface pressure of the body and the volume rendering of absolute value of the vorticity.

The number of the grid point is 288*144*144=5,971,968 in the case of sphere and 256*128*128=4,194,304 in the case of protoceratopus. These numbers are not enough to see the smallest structures but look to be good enough to see the large structure in the transitional stage.

In this computation, only cartesian grid with equal grid spacing is used. Body fitting type grid system is not only difficult to make but uniformity of the grid is not possible. To see the behavior of the transition to turbulence, this uniformity is crucial. The visualization software used here is Clef2D and Clef3dvr developed by Institute of Computation Fluid Dynamics, (Kuzuu, Kaizaki, and Kuwahara 1997).

1) Transitional flow past a sphere

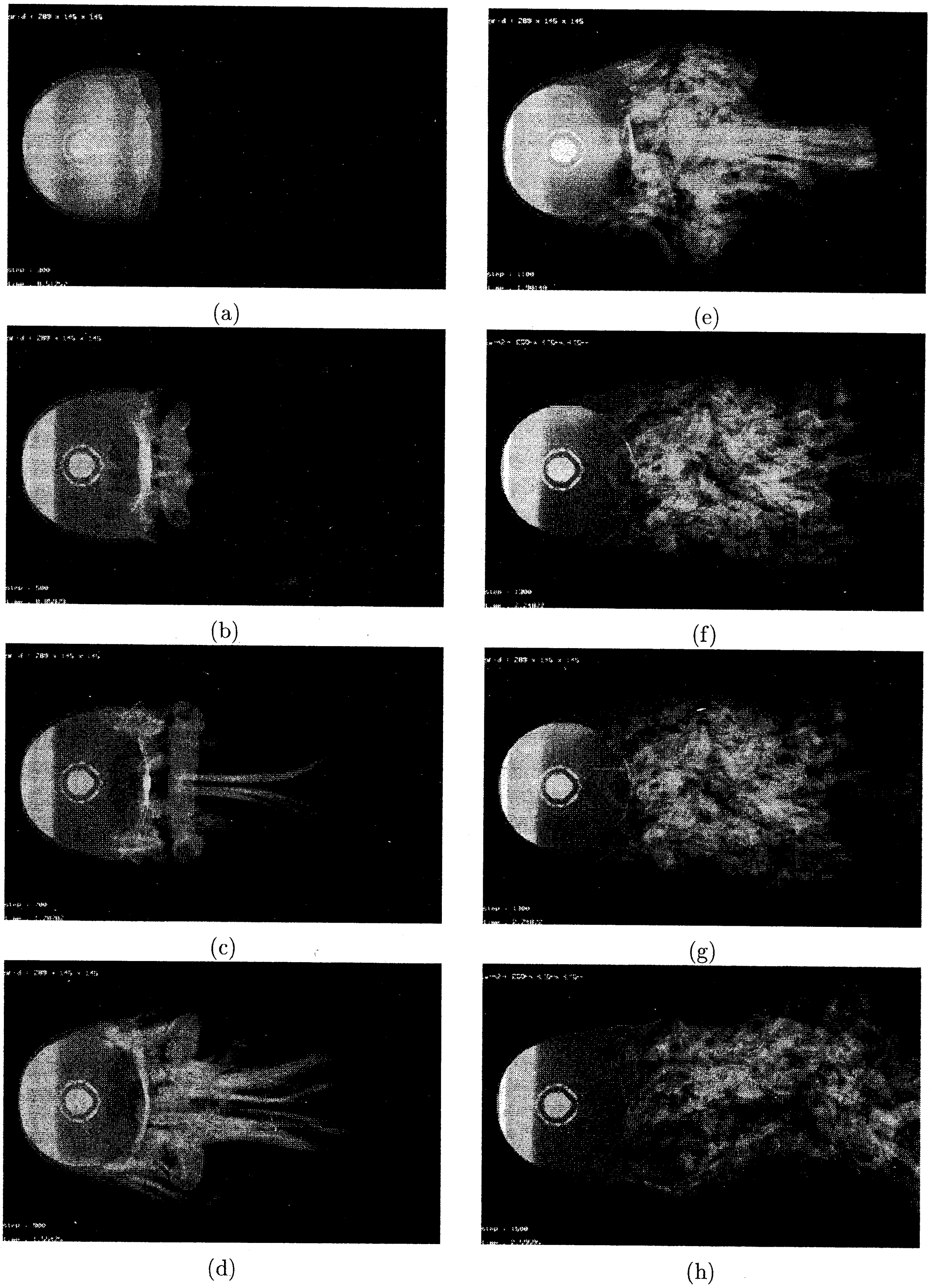


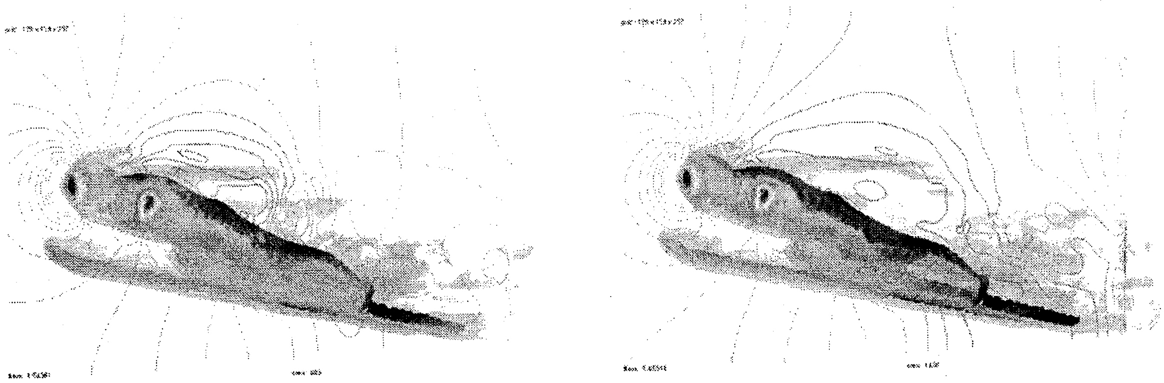

**Fig.1.** Development of turbulence behind a sphere at Re=200,000, 288*144*144 grid

2) Front view

(a) Just after the start (b) Fully developed stage

**Fig.2.** Flow around a ski jumper at Re=200,000, 288*144*144 grid

3) Side view



(a) Just after the start (b) Fully developed stage

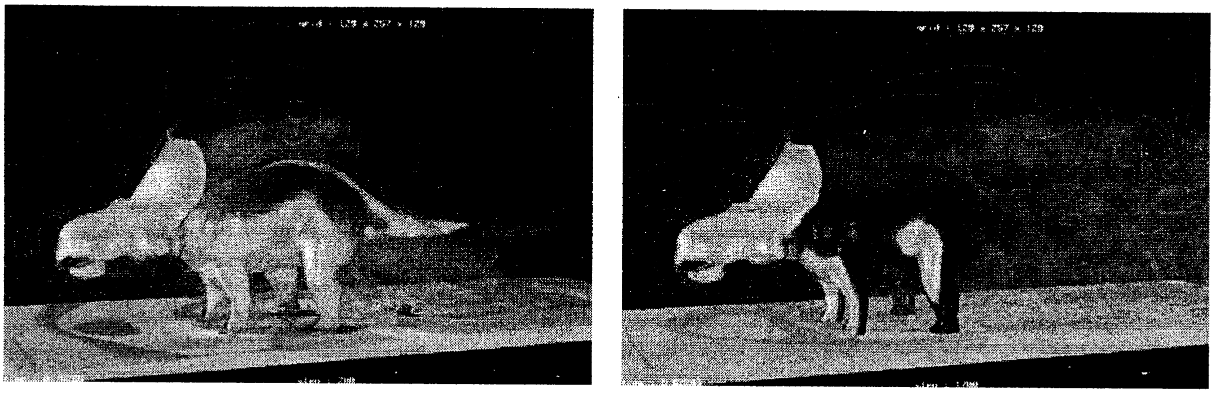
**Fig.3.** Flow around a ski jumper at Re=200,000, 288*144*144 grid

4) Rear view

(a) Just after the start (b) Fully developed stage

**Fig.4.** Flow around a ski jumper at Re=200,000, 288*144*144 grid

5) Flow around a protoceratopus



(a) Initial stage (b) Fully developed stage

**Fig.5.** Flow around a protoceratopus At Re=100,000, The grid is 256*128*128

## IV. Conclusions

It is becoming clear that we need not resolve the small-scale structure of high-Reynolds-number flow to capture the large structure, which is most important for application. We should not use standard models to simulate any high-Reynolds-number, turbulent flows. Only without using turbulence models we are able to capture the dependence of the flow on the Reynolds number. To avoid the numerical instability we can simply use a third-order upwind difference. Multi-directional finite-difference makes the dependence of the solution on the flow direction less and the computation more reliable.

With these considerations, transition to turbulence can be well simulated, and the structure is self explanatory with proper visualization.

## References


1) Kawamura, T., and Kuwahara, K., 1984, "Computation of high Reynolds number flow around a circular cylinder with surface roughness", AIAA Paper 84-0340.
2) Kuzuu, K., and Kaizaki, H., and Kuwahara, K., 1997, "Real Time Visualization of Flow Fields Using Open GL System", AIAA Paper 97-0235.
3) Kuwahara, K., 1991, "Flow Simulation on Supercomputers and Its Visualization", International Journal of High Speed Computing, Vol.4, No.1, pp49-70.